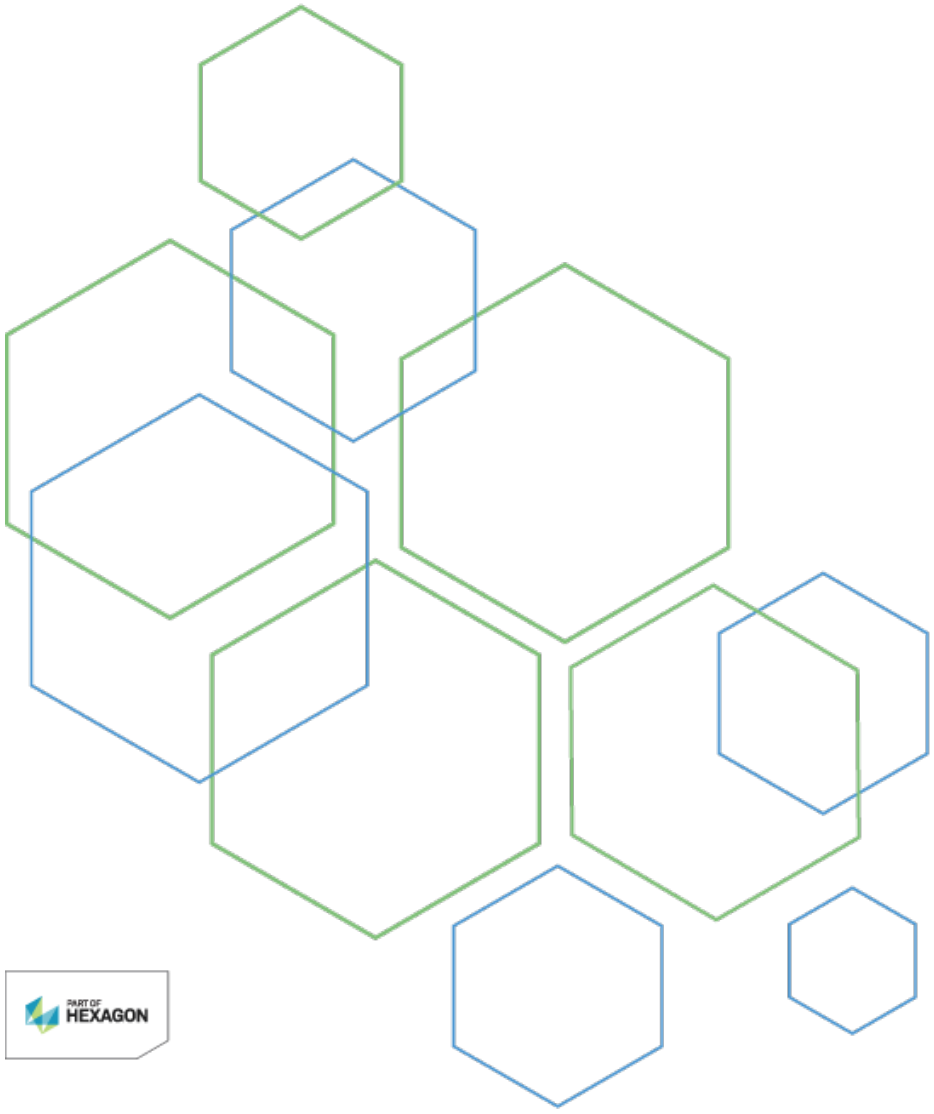
INTERGRAPH Smart 3D

Geometry Analysis and Repair

User's Guide



PART OF HEXAGON

Version 2016 (11.0)
November 2016

Copyright

U.S. Government Restricted Rights Legend

Intergraph Corporation
305 Intergraph Way
Madison, AL 35758

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on Intergraph Smart Support, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

Terms of Use

a. Use of a software product and Documentation is subject to the End User License Agreement ("EULA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.

b. For use of Documentation or Other Documentation where end user does not receive a EULA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee's internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee's internal, non-commercial use. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

Except for any express warranties as may be stated in the EULA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software, Documentation and Other Documentation discussed in this document are furnished under a license and may be used or copied only in accordance with the terms of this license. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

Limitation of Damages

IN NO EVENT WILL INTERGRAPH CORPORATION BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL INCIDENTAL, SPECIAL, OR PUNITIVE DAMAGES, INCLUDING BUT NOT LIMITED TO, LOSS OF USE OR PRODUCTION, LOSS OF REVENUE OR PROFIT, LOSS OF DATA, OR CLAIMS OF THIRD PARTIES, EVEN IF INTERGRAPH CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

UNDER NO CIRCUMSTANCES SHALL INTERGRAPH CORPORATION'S LIABILITY EXCEED THE AMOUNT THAT INTERGRAPH CORPORATION HAS BEEN PAID BY LICENSEE UNDER THIS AGREEMENT AT THE TIME THE CLAIM IS MADE. EXCEPT WHERE PROHIBITED BY APPLICABLE LAW, NO CLAIM, REGARDLESS OF FORM, ARISING OUT OF OR IN CONNECTION WITH THE SUBJECT MATTER OF THIS DOCUMENT MAY BE BROUGHT BY LICENSEE MORE THAN TWO (2) YEARS AFTER THE EVENT GIVING RISE TO THE CAUSE OF ACTION HAS OCCURRED.

IF UNDER THE LAW RULED APPLICABLE ANY PART OF THIS SECTION IS INVALID, THEN INTERGRAPH LIMITS ITS LIABILITY TO THE MAXIMUM EXTENT ALLOWED BY SAID LAW.

Export Controls

Intergraph Corporation's software products and any third-party Software Products obtained from Intergraph Corporation, its subsidiaries, or distributors (including any Documentation, Other Documentation or technical data related to these products) are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. These Software Products, and the direct product thereof, must not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

a. To Cuba, Iran, North Korea, Sudan, or Syria, or any national of these countries.

b. To any person or entity listed on any U.S. government denial list, including but not limited to, the U.S. Department of Commerce Denied Persons, Entities, and Unverified Lists, http://www.bis.doc.gov/complianceandenforcement/liststocheck.htm, the U.S. Department of Treasury Specially Designated Nationals List, http://www.treas.gov/offices/enforcement/ofac/, and the U.S. Department of State Debarred List, http://www.pmddtc.state.gov/compliance/debar.html.

c. To any entity when Licensee knows, or has reason to know, the end use of the Software Product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.

d. To any entity when Licensee knows, or has reason to know, that an illegal reshipment will take place.

Any questions regarding export or re-export of these Software Products should be addressed to Intergraph Corporation's Export Compliance Department, Huntsville, Alabama 35894, USA.

Trademarks

Contents

Preface

This document is a user's guide for the Geometry Analysis and Repair functionality of Intergraph Smart™ 3D and provides command reference information and procedural instructions.

Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support* (*https://smartsupport.intergraph.com*).

What's New in Geometry Analysis and Repair

The following changes have been made to the Geometry Analysis and Repair task.

Version 2016 (11.0)

- Added a new locate filter, **Construction Graphics**. For more information, see *Displaying Properties* (on page 34). (P2 CP:271166).

SECTION 1

Geometry Analysis and Repair

The Geometry Analysis and Repair task of Intergraph Smart™ 3D allows you to analyze, check, and repair geometry data before using that data to create an imported plate system in the Molded Forms task. You can run the Geometry Analysis and Repair commands on data that will be used in "early design" activities as well as data that will be used to detail and manufacture the model.

This task is different from other Smart 3D tasks in that it does not create or save objects in the database. You should not refresh your workspace unless you want to clear the screen. The **Workspace Explorer** does not appear in this task, and there are no modify commands in this task.

You start the Geometry Analysis and Repair task by clicking **Tasks** > **Geometry Analysis and Repair**. The Geometry Analysis and Repair task has the following commands:

Select - Used to select objects. For more information, see *Displaying Properties* (on page 34).

Process Geometry Data - Checks and repairs data. For more information, see *Process Geometry Data* (on page 17).

Reference Curve - Creates reference curves. For more information, see *Reference Curves* (on page 24).

Advanced Surface Operations - Performs operations on surfaces such as stitching, exploding, splitting, rebuilding, and merging data. For more information, see *Advanced Surface Operations* (on page 26).

Advanced Curve Operations - Performs operations on curves such as splitting, merging, rebuilding, and exploding. For more information, see *Advanced Curve Operations* (on page 27).

Transform Data - Scales, translates, rotates, and mirrors data. For more information, see *Transform Data* (on page 30).

Compare Data - Compares surface data before and after correction to ensure that the differences are within acceptable tolerances of the original data. For more information, see *Compare Data* (on page 32).

Import IGES - Imports 3D geometric data from a file in the IGES (Initial Graphics Exchange Standard) format. This command is on the **File** menu. For more information, see *Import IGES (File Menu)* (on page 14).

Import ACIS - Imports 3D geometric data from a file in the ACIS .sat format. This command is on the **File** menu. For more information, see *Import ACIS (File Menu)* (on page 14).

Export IGES - Exports 3D geometric data to a file in the IGES format. This command is on the **File** menu. For more information, see *Export IGES (File Menu)* (on page 16).

Export ACIS - Exports geometric data to a file in the ACIS .sat format. This command is on the **File** menu. For more information, see *Export ACIS (File Menu)* (on page 16).

Geometry Analysis and Repair Setup - Specifies settings for the Geometry Analysis and Repair task, including logging, display, and style options. This command is on the **Tools** menu. For more information, see *Geometry Analysis and Repair Setup Command (Tools Menu)* (on page 11).

Show Control Points - Displays the control points and the control point mesh for the selected data. This command is on the shortcut menu available when no command is active and you right-click a graphic object in this task. For more information, see *Show Control Points* (on page 36).

Show Normals - Displays the surface normals for the selected data. This command is on the shortcut menu available when no command is active and you right-click a graphic object in this task. For more information, see *Show Normals* (on page 37).

Show Vertices - Displays vertices for the selected data. A vertex is a point in three- dimensional space. This command is on the shortcut menu available when no command is active and you right-click a graphic object in this task. For more information, see *Show Vertices* (on page 37).

Hide - Removes all control points, normals, and vertex data from the display. This command is on the shortcut menu available when no command is active and you right-click a graphic object in this task. For more information, see *Hide* (on page 38).

SECTION 2

Understanding the Geometry Analysis and Repair Workflow

Using the Geometry Analysis and Repair task, you can complete the following operations:

- Specify settings for symmetry, logging of operations, and styles in this task.
- Import and export geometric data.
- Check the data against a defined set of requirements.
- Get information on data that does not pass the check.
- Repair problems in the data.
- Save the results of automatic and manual repairs to an external data file.

After analyzing and repairing the geometry data, you can import it into Molded Forms with the **Imported Plate System** command. For more information, see the Molded Forms documentation.

See Also

Geometry Analysis and Repair Common Tasks (on page 9)

Geometry Analysis and Repair Common Tasks

The following tasks are used frequently in the Geometry Analysis and Repair task.

Specify Setup Options

You can set options for symmetry, logging, and styles in the Geometry Analysis and Repair task. For more information, see *Specify Settings for Geometry Analysis and Repair* (on page 13).

Import Data

You can import geometry data in ACIS or IGES format. For more information, see *Import ACIS or IGES Data* (on page 15).

Process Data against Requirements

The primary command in the Geometry Analysis and Repair task is the **Process Geometry Data** command, which checks and repairs data. For more information, see *Process Data for Geometry Analysis and Repair* (on page 18).

Perform Other Operations on the Data

You might need to rotate and orient the data to meet the software's requirements. For more information, see *Transform Data* (on page 30).

You can also perform operations that are not automatically done as part of the repair process. These operations result in higher quality data, smaller data sets, and better conformance. For

more information, see *Advanced Surface Operations* (on page 26) and *Perform Advanced Curve Operations* (on page 28).

It can be helpful to compare modified data with baseline data when deciding if and how to change the data. For more information, see *Compare Data* (on page 32).

Save and Export Data

After repairing the data, you must save it to an external file. In the Geometry Analysis and Repair task, no objects or changes are saved to the database. For more information, see *Export ACIS and IGES Data* (on page 16).

SECTION 3

Setting Options for the Geometry Analysis and Repair Task

The Geometry Analysis and Repair task provides some overall options for you to set, including options for the log file and the appearance of objects. You can set or modify these options at any time when working in this task.

See Also
Specify Settings for Geometry Analysis and Repair (on page 13)

Geometry Analysis and Repair Setup Command (Tools Menu)

Displays a dialog box that allows you to set options for the Geometry Analysis and Repair task.

For more information, see *Geometry Analysis and Repair Setup Dialog Box* (on page 11).

Geometry Analysis and Repair Setup Dialog Box

Specifies settings for the Geometry Analysis and Repair task.

Options Tab

Specifies various options for the Geometry Analysis and Repair task.

Symmetrical Hull

Specifies that the hull is symmetrical. If checked, the software strips off the starboard side of the ship, and all operations are performed on the port side. By default, the **Symmetrical Hull** option is checked.

If the **Symmetrical Hull** option is selected, you will be able to copy and mirror the geometry at the end of the **Process Geometry Data** command by selecting the **Copy/Mirror about Centerline** option on the *Process Settings dialog box* (on page 23). The software does not copy and mirror geometry that crosses the centerline.

Activate Logging

Activates logging of all operations in this task. The logging file contains a running list of objects that are processed. Each time you open the task, information is appended to this file. To clear the file, you can open it and delete the contents, or you can specify a new file.

Log File Save Location (.log)

Click the **Browse** button to designate a log file name and location.

Styles Tab

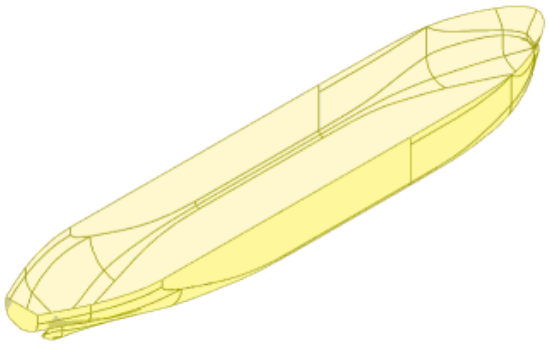
Specifies the styles for normal state objects as well as objects in error in the Geometry Analysis and Repair task.

The styles on this tab are saved when you save your session file.

Bodies

Specifies the style for bodies in a normal state.

The following picture shows a sheet-body with the translucent yellow style. To see the edges clearly (as in the picture), you can click **Format > View** and change the rendering mode to **Shaded with Enhanced Edges**.



Faces

Specifies the style for faces in a normal state.

Edges

Specifies the style for edges in a normal state.

Vertices

Specifies the style for vertices in a normal state.

Face Errors

Specifies the style for faces in error.

The following picture shows a face in error with the red style.

Edge Errors

Specifies the style for edges in error.

Vertex Errors

Specifies the style for vertices in error.

NOTE To see through objects, you can assign translucent styles. For example, it is often helpful to be able to see through body and face errors.

Specify Settings for Geometry Analysis and Repair

1. Click **Tools > Geometry Analysis and Repair Setup**.
2. On the **Options** tab, select the **Symmetrical Hull** option, if the hull is symmetrical.

   TIPS

   - If this option is checked, the software strips off the starboard side of the ship, and operations are performed on the port side.
   - You can copy and mirror the geometry at the end of the **Process Geometry Data** command by selecting the **Copy/Mirror about Centerline** option on the *Process Settings dialog box* (on page 23). The software does not copy and mirror any objects that cross the centerline.
3. Activate and specify a log file, if logging is necessary.

   TIP The logging file contains a running list of objects that are processed. Each time you open the task, information is appended to this file. To clear the file, open it and delete the contents, or specify a new file.
4. On the **Styles** tab, specify the styles for objects in their normal state and for objects in error.

TIPS

- You can assign translucent styles to body and face errors, for example, in order to see through them.
- To see edges more clearly, you can click **Format > View** and change the rendering mode to **Shaded with Enhanced Edges**.

SECTION 4

Importing ACIS and IGES Data

You can temporarily import geometry data in ACIS or IGES format from other software packages, such as Tribon, NAPA, or Rhino. You can compare the temporary geometry to the geometry of objects in the Model.

During the import process, the software applies LGF attributes to patches, if they are present. This information is used later during a copy and mirror operation, if the hull is symmetrical.

You can also export geometry data in ACIS or IGES format. The export operation operates against selected sheetbody and wirebody objects.

If you are exporting in IGES format, you can set the output data units using the **Tools > Options** command.

IMPORTANT

- You must have a license for IGES in order to use the **Import > IGES** or **Export > IGES** command.
- The data in the Geometry Analysis and Repair task is not saved in a session file or in the database. If you delete geometry, or define or refresh the workspace, the imported geometry is lost. You must export the data to an external file to save it. The imported geometry cannot be used to create objects in the database.

The Import and Export ACIS/IGES commands are available in the Molded Forms and Geometry Analysis and Repair tasks. The Structural Detailing task allows you to export ACIS and IGES data.

Import ACIS (File Menu)

File > **Import** > **ACIS** imports wireframe and surface data from a file in the Acis .sat format. The imported geometry is not saved in the session file or in the database. After processing the data, you will need to export it to save your work. For more information, see *Export ACIS (File Menu)* (on page 16).

Import IGES (File Menu)

File > **Import** > **IGES** imports wireframe and surface geometry data from a file in IGES format into the software. The imported geometry is not saved in the session file or in the database. After processing the data, you will need to export it to save your work. For more information, see *Export IGES (File Menu)* (on page 16).

NOTE You must have a license for IGES in order to use this command.

Import ACIS or IGES data

1. Click **File** > **Import** > **ACIS** or **File** > **Import** > **IGES**, depending on what type of data to import.
2. Browse to locate the file.
3. Click **Open**.

NOTES

- A log file is saved in the folder where the input file resides.
- You can import data over other data. The effect is cumulative.

SECTION 5

Exporting ACIS and IGES Data

You can export geometry data in ACIS or IGES format. The export operation operates against selected sheet-body and wire-body objects.

If you are exporting in IGES format, you can set the output data units using the **Tools > Options** command. For more information about **Tools > Options**, see the *Common User's Guide*.

For more information about exporting ACIS and IGES data, see *Export ACIS (File Menu)* (on page 16) and *Export IGES (File Menu)* (on page 16).

IMPORTANT

- You must have a license for IGES in order to use the **Export** > **IGES** command.
- The data in the Geometry Analysis and Repair task is not saved in a session file or in the database. If you delete geometry, or define or refresh the workspace, the imported geometry is lost. You must export the data to an external file to save it.

Export ACIS (File Menu)

File > Export > ACIS saves the selected wireframe and surface geometry data to a file in ACIS format. This command supports the output of geometry to a file but not object properties.

Export IGES (File Menu)

File > Export > IGES saves the selected wireframe and surface geometry data to a file in IGES format. This command supports the output of geometry to a file but not object properties. You must have a license for IGES in order to use this command.

Export ACIS and IGES Data

1. Select the data that you want to export.
2. Click **File > Export > ACIS** or **File > Export > IGES**, depending on what type of data to export.

   TIP You must have an IGES license in order to export IGES data.
3. Browse to locate the file.
4. Click **Save**.

NOTE A log file is saved in the folder where the export file resides.

SECTION 6

Processing Data

The main command in the Geometry Analysis and Repair task is the **Process Geometry Data** command, which provides basic checking and repairing processes. Further processing can be done using the other commands in this task.

When you use this command, you first set some options, such as the check level, save version, and tolerances for points, knuckles, and tangents. Then, you run the checking process. The software notifies you of the problems that it finds. You can repair the data and run the checks again. Some repairs can be automatically fixed by the software, and other repairs are more complex and will require more manual fixes. The last option in the **Process Geometry Data** command is saving the data to an export file.

Process Geometry Data

 Processes hullform data. This command allows you to modify default options, select the objects to process, check the objects, repair the objects with problems, and save the data to a file.

Process Geometry Data Ribbon

Sets options for processing geometry data.

 **Options**

Displays the **Options** dialog box, which allows you to specify settings such as the check level, save version, and tolerances. For more information, see *Options Dialog Box* (on page 19).

 **Select**

Allows you to select sheet-bodies or wire-bodies in the model.

 **Check**

Displays the **Check Settings** dialog box, which allows you to choose which checks to perform. For more information, see *Check Settings Dialog Box* (on page 20).

 **Repair**

Displays the **Repair Settings** dialog box, which allows you to choose which repairs to perform. For more information, see *Repair Settings Dialog Box* (on page 22).

 **Process**

Displays the **Process Settings** dialog box, which allows you to choose which processes to perform. For more information, see *Process Settings Dialog Box* (on page 23).

 **Save Changes**

Saves the sheet-body that was processed and all reference curves loaded in the view. The data is saved in the version specified on the *Options Dialog Box* (on page 19).

Process Data for Geometry Analysis and Repair

1. Click **Process Geometry Data** on the vertical toolbar.
2. On the ribbon, click **Options** to specify the options for this command.

   Set Options for Processing Data (on page 18)
3. Select the objects to process.

   TIP You can select only part of the hull to process at a time if you want.
4. On the ribbon, click **Check** to specify the check settings, and run the check process.

   Check Data (on page 18)
5. Click **Repair** to specify the repair settings, and run the repair process.

   Repair Data (on page 19)
6. After checking and repairing all the data, click **Process** to specify the process settings, and run the process.

   Process Data (on page 19)
7. Click **Save Changes** if you want to save the data in .sat file format. The software saves the sheetbody that you processed along with all the reference curves loaded in the view.

   TIP The data is saved in the version specified in the **Acis Save Version** box on the *Options dialog box* (on page 19).

NOTE You should repeat the steps in this procedure as many times as needed.

Set Options for Processing Data

1. In the **Acis Check Level** box, select the level for the checks to be performed. Level 20 is the most basic, and Level 70 is the most thorough.
2. In the **Acis Save Version** box, select the version for the .sat file saved after running the **Process Geometry Data** command.
3. In the **Points** box, type the tolerance value that will determine coincident points.
4. In the **Knuckles** box, type the tolerance value that will determine knuckled edges.
5. In the **Tangents** box, type the tolerance value that will determine smooth edges.

NOTE You can specify the precision for the tolerances by opening the **Tools > Options** dialog box and clicking the **Units of Measure** tab.

Check Data

1. On the **Check Settings** dialog box, select the checks that you want to run.

   TIP The checks above the line on the dialog box have repairs that the software can complete automatically. The checks below the line on the dialog box have repairs that you must complete manually.

2. Click **Execute**.

NOTES

- The software displays ✖ on the **Check Settings** dialog box for the checks that fail. In addition, the geometry that failed is highlighted in red in the graphic view. You can select an object, and click **Properties** on the ribbon to see more information about the errors. You can also open the log file to view more details about the checks and errors.
- The software displays ✓ on the **Check Settings** dialog box for the checks that succeed.

Repair Data

1. On the **Repair Settings** dialog box, select the repairs that you want to run.

   NOTE The software automatically enables and selects the repairs for the checks that failed during the **Check** step of the **Process Geometry Data** command. You can clear these repairs, if needed.

2. Click **Execute**.

NOTES

- The software displays ✖ on the **Repair Settings** dialog box for the repairs that fail. The software displays ✓ on the **Repair Settings** dialog box for the repairs that succeed.
- Typically, you want to check and repair the data until every setting displays ✓. You can then proceed to *Process Data* (on page 19).

Process Data

1. On the **Process Settings** dialog box, select the operations that you want the software to do during processing.

   TIP The available operations are applying knuckle attributes, copying and mirroring about the centerline, and cleaning the sheet-body of attributes that are no longer needed. These operations prepare the data for export.

2. Click **Execute**.

Options Dialog Box

Provides settings for the **Process Geometry Data** command.

ACIS Check Level

Specifies the level of checks to be performed. Level 20 is basic checking, while level 70 is the most thorough. For more information about the checks at these levels, see the ACIS documentation.

ACIS Save Version

Specifies the version of the .sat file that will be saved. ACIS supports backward compatibility, and this function allows data created on a later version to be usable by an earlier version.

Tolerances

Allows you to specify permissible deviations for the definitions of points, knuckles, and

tangents.

NOTE You can specify the precision for the tolerances by opening the **Tools > Options** dialog box and clicking the **Units of Measure** tab. For more information about **Tools > Options**, see the *Common User's Guide*.

Points

Specifies the value used to determine coincidence. If the distance between two points is less than this value, the software assumes they are the same point. The default **Points** tolerance is 0.001 m.

Knuckles

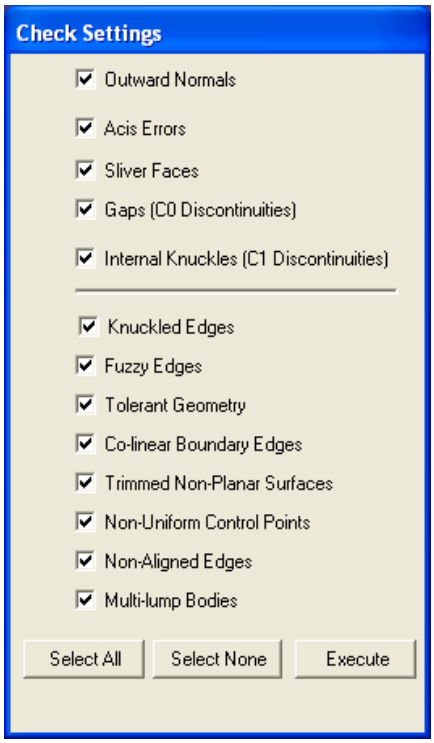
Specifies the value used to determine whether tangent discontinuities across face edges are discontinuous (knuckled). The default **Knuckles** tolerance is 2.000 deg.

Tangents

Specifies the value used to determine whether tangent discontinuities across face edges are continuous (smooth). The default **Tangents** tolerance is 0.010 deg.

Check Settings Dialog Box

Provides settings for the **Check** portion of the **Process Geometry Data** command.



The checks above the line on this dialog box all have automatic repairs that can be run to correct the problematic data. The software selects the appropriate repair when an object fails a check. However, you can choose to override the selection and skip the repair.

Outward Normals

Checks the selected objects for outward surface normals.

Acis Errors

Checks the selected objects with the ACIS- supplied functions. The checks are performed at the level defined by the **Acis Check Level** setting on the *Options Dialog Box* (on page 19).

Sliver Faces

Checks the selected objects for sliver faces. This check ensures that face edges are separated from one another by more than the **Points** tolerance specified on the *Options Dialog Box* (on page 19).

Any face can have one or two degenerate edges, but in general, edges must be distinguishable from one another, or stitching errors can occur.

Gaps (C0 Discontinuities)

Checks the selected objects for gaps (ACIS C0 discontinuities). This check ensures that adjacent patch edges are within ACIS' tolerances for coincidence. If this option is selected, the checks are performed using the tolerance defined by the **Points** setting on the *Options Dialog Box* (on page 19).

Internal Knuckles (C1 Discontinuities)

Checks the selected objects for internal knuckles. This check ensures that there are no ACIS C_1 discontinuities within any of the faces of the body.

The checks below the line on this dialog box do NOT have automatic repairs that can be run to correct the problematic data. They must be manually repaired either in the originating third-party application that created the data or by using one or more of the **Advanced Surface Operations** or **Advanced Curve Operations** commands. For more information, see *Performing Advanced Operations* (on page 26).

Knuckled Edges

Checks the selected objects for tangent discontinuities across face boundaries that exceed the tolerance specified in the **Knuckle** box on the *Options Dialog Box* (on page 19).

Fuzzy Edges

Checks the selected objects for tangent discontinuities across face boundaries that are greater than the tolerance specified in the **Tangents** box but less than the tolerance specified in the **Knuckles** box on the *Options Dialog Box* (on page 19). Objects that fail this check (exceed this tolerance) are displayed in red.

Tolerant Geometry

Checks the selected objects for the presence of tolerant edges and/or vertices.

Co-linear Boundary Edges

Checks the selected objects to ensure that none of the face edges in the body have co-linear tangents.

Trimmed Non-Planar Surfaces

Checks the selected objects for "trim" boundaries, which reduce the active region of the surface from its natural boundaries.

Non-Uniform Control Points

Checks the selected objects to ensure that there are an equal number of control points along the common edges of adjacent faces.

Non-Aligned Edges

Checks the selected objects to ensure that adjacent face edges are aligned along the common boundary.

Multi-lump Bodies

Checks the selected objects to see if they are ACIS bodies with more than one lump (disjoint geometry objects).

Select All

Selects all checks.

Select None

Clears all checks.

Execute

Runs the check process.

Repair Settings Dialog Box

Provides settings for the **Repair** portion of the **Process Geometry Data** command.

These settings are not saved with the preferences; rather, they are automatically set based on the results of the checks. You can bypass a repair by clearing the check box for the repair. If all the checks are successful, then the **Repair** step of the **Process Geometry Data** command is not available.

Orient Normals

Forces the normals to be consistently oriented inwards.

Slivers

Removes the faces with edges completely within the **Points** tolerance of each other. The **Points** tolerance is set on the *Options Dialog Box* (on page 19).

Gaps

Closes the gaps between adjacent face edges by adjusting the control points along one of the edges to match the control points along the other edge.

Triangles

Fixes the triangular faces formed with degenerate edges defined by coincident control points.

Internal Knuckles

Fixes the faces with internal knuckles (C_1 discontinuities) by splitting at the knuckle.

Convexities

Splits the edges of the selected objects at changes in convexity points.

Execute

Runs the repair process.

Process Settings Dialog Box

Provides settings for the **Process** portion of the **Process Geometry Data** command.

Apply Knuckle Attribute

Specifies whether the knuckle edge attributes are added to the sheet-body during processing. If this box is checked, any existing attributes are first removed before the new attributes are applied.

Copy/Mirror about Centerline

Specifies that all of the port faces in the sheet-body are copied and mirrored about the centerline (Y=0 in the global coordinate system).

NOTE This option is selected by default if the **Symmetrical Hull** option is selected on the *Options tab* (on page 11) of the **Geometry Analysis and Repair Setup** dialog box. The option is not available if the **Symmetrical Hull** option is not selected.

Clean

Specifies whether the attributes applied in the **Process Geometry Data** command are removed from the sheet-body.

Execute

Runs the process.

SECTION 7

Reference Curves

Assigns names and other properties to knuckle, tangent, and reference curves. In the Geometry Analysis and Repair task, you can quickly and easily place attributes on knuckle, tangent, and reference curves using the **Reference Curves** command.

You use the **Reference Curves** command with the **Advanced Curve Operations** command. First, you create the curves with a merge operation in the **Advanced Curve Operations** command. Then, you assign properties to the curves using the **Reference Curves** command.

Reference Curves Ribbon

Sets options for placing attributes on a knuckle, tangent, or reference curve.

Select

Select a single wire-body from a graphic view. The existing **Name**, **Symmetry**, and **Type** values appear in their respective boxes when you select the wire-body.

Apply Attributes

Adds the specified properties to the selected object.

Undo

Reverses the operation by replacing the results with the data before the operation was applied. You can modify a setting, re-apply, undo, and repeat until you are satisfied with the results.

Name

Defines a prefix for the wire-body. The complete name is this text value appended by a "_" and the symmetry value: P for Port, S for Starboard, or C for Centered.

Symmetry

Specifies the symmetry to append to the reference curve name.

If the symmetry value is **None**, the software does not append any value to the name.

If the symmetry value is **Port/Starboard**, the selected wire-body is named **Port**, and a copy is created and mirrored about the centerline and assigned a **Starboard** symmetry value.

Type

Specifies the type of reference curve: a general purpose reference curve, a tangent curve, or a knuckle curve.

How to review Reference Curves

1. Click **Reference Curves**.
2. Select a wire-body in a graphic view.

3. Check the name, symmetry, and type values on the ribbon, and change them, if necessary.

   TIP If you choose **Port/Starboard** in the **Symmetry** box, the software creates the symmetric curve for you.

4. Click **Apply Attributes** .

NOTE You can revert back to the original values on a reference curve by clicking **Undo** on the ribbon before applying the change. The **Name**, **Symmetry**, and **Type** properties are reset. The software also deletes the symmetrical starboard copy, if it was created.

SECTION 8

Performing Advanced Operations

The Geometry Analysis and Repair task allows you to perform a number of advanced operations on surfaces (faces) and on curves (edges) including: splitting, merging, stitching, exploding, rebuilding, and snapping.

These operations are optional and are not automatically done as part of the repair process and vary on a case-by-case basis depending on the initial data. They can result in higher quality data, smaller data sets, and better conformance to Smart 3D requirements.

★IMPORTANT The advanced operations have the potential to change the accuracy of the data; therefore, it is recommended to be trained in their usage and to exercise care and caution when using these operations. You must carefully analyze and check the data produced and altered by these functions before using the data in production.

Advanced Surface Operations

 Performs advanced operations on surfaces (faces). These operations are not automatically done as part of the repair process.

Advanced Surface Operations Ribbon

Sets options for performing advanced surface operations.

Operation

Select the operation to perform. The operation remains selected until you choose another.

- **Split** - You select a single face to be split. Then, you select a single intersecting edge to be the splitter.
-  **Select the Splitter** - Select an intersecting edge, or the point for the intersecting edge. This option is only active when the **Split** operation is selected.
- **Merge** - You select any combination of sheet-bodies and faces. The software attempts to merge them together into a new sheet-body.
- **Merge Tolerance** - Value that controls the creation of tolerant geometry during the merge process.
- **Explode** - You select sheet-bodies, and the software converts them to individual, disconnected, faces.
- **Rebuild** - You select one or more faces. The software creates new geometry by rebuilding the underlying surfaces with the specified degree and number of control points. You can specify that trimmed faces are converted to natural faces with the **Trim to Boundaries** option.
- **Snap** - You select any number of faces. The software adjusts the faces that lie within the specified tolerance to lie exactly on the plane. You can choose an object in the view to set the plane.

Select

Selects one or more objects from the view for the chosen operation.

Apply

Applies the operation to the selected objects.

Undo

Reverses the operation by replacing the results with the data before the operation was applied. You can modify a setting, re-apply, undo, and repeat until you are satisfied with the results.

How to do Advanced Surface Operations

1. Click **Advanced Surface Operations** .
2. In the **Operation** box, select an operation to perform.
3. Click **Apply** .

NOTE You can use **Undo** on the ribbon to reverse an operation without exiting the command. For example, you can modify a setting, re-apply, undo, and repeat until you are satisfied with the results.

Advanced Curve Operations

Performs advanced operations on curves (edges). These operations are not automatically done as part of the repair process.

Advanced Curve Operations Ribbon

Sets options for performing advanced curve operations.

Operation

Select the operation to perform. The current operation remains active until you choose another.

- **Merge** - You select a connected set of edges. The software merges them into a single wire-body, providing they are endpoint- connected within the specified merge tolerance.
- **Chain** - If the select set contains a single edge, then this button controls the automatic selection of connected edges based on the starting and ending tangents. Edges are automatically added to the select set as long as the tangent difference between the edges is less than the tangent threshold. If multiple edges extend from the end of an edge, the edge with the smallest tangent difference is used. The chain stops when no edge is found with a tangent less than or equal to the threshold.
- **Merge Tolerance** - Value that specifies how close the edges must be at their endpoints to be included.
- **Tangent Threshold** Value that specifies whether or not an edge is added during the chaining operation.
- **Split** - You select a single wire-body to be split. Then, you select an intersecting edge to be the splitter.

- **Select the Splitter** - Select an intersecting edge. This option is only active when the **Split** operation is selected.
- **Explode** - You select wirebodies, and the software converts them to individual, disconnected edges.
- **Rebuild** - You select one or more edges. The software creates new geometry by rebuilding the underlying curves with the specified degree and number of control points.
- **Snap** - You select any number of edges. The software adjusts the edges that lie within the specified tolerance to lie exactly on the plane. You can choose an object in the view to set the plane.

Select

Selects one or more objects from the view for the chosen operation.

Apply

Applies the operation to the selected objects.

Undo

Reverses the operation by replacing the results with the data before the operation was applied. You can modify a setting, re-apply, undo, and repeat until you are satisfied with the results.

Perform Advanced Curve Operations

1. Click **Advanced Curve Operations** on the vertical toolbar.
2. In the **Operation** box, select an operation to perform.

   Merge (on page 28)
   Split (on page 29)
   Explode (on page 29)
   Rebuild (on page 29)
   Snap (on page 29)

3. Click **Apply** .

NOTE You can use **Undo** on the ribbon to reverse an operation without exiting the command. For example, you can modify a setting, re- apply, undo, and repeat until you are satisfied with the results.

Merge

1. In the graphic view, select one or more edges in a connected set of edges.

   TIP You can select the edges one by one, or you can use **Chain** on the ribbon to automatically select all connected edges based on the Merge Tolerance and **Tangent Threshold** settings.

2. In the **Merge Tolerance** box on the ribbon, select or type a value. This value controls how close the endpoints of the edges must be.
3. In the **Tangent Threshold** box, select or type a value. This value controls the chaining operation.

Split

1. In the graphic view, select a wire-body to split.
2. Select an intersecting edge to be the splitter.

Explode

- In a graphic view, select wire-bodies. The **Explode** operation converts these wire-bodies to individual edges.

Rebuild

1. Select one or more edges in the graphic view.
2. In the **Degree U** box, select or type a value for the degree of the curve.
3. In the **Num U** box, select or type a value for the number of control points on the curve.

Snap

1. Select edges in the graphic view.
2. On the ribbon, specify a plane to snap to in the **Plane** box.
3. Type a number in the **Value** box.

   TIP Click **Pick** to choose an object in the view that specifies this value.
4. Type a number in the **Tolerance** box. This number provides some leeway around the specified plane and value.

SECTION 9

Transform Data

Scales, translates, rotates, or mirrors selected objects. You might encounter situations where the imported data is not located or oriented in accordance with the requirements of Smart 3D. In the Geometry Analysis and Repair task, you can perform basic scaling, translation, rotation, and mirroring of the geometry in the global coordinate system.

Transform Data Ribbon

Sets options for transforming data.

Select

Selects objects to transform.

Transform

Specifies the transform operation.

- **Scale** - Changes the scale of the selected objects.
- **Scale Type** - Specifies the type of scaling, **Uniform** or **Non-Uniform**.
- **Scale Factor** - For uniform scaling, specifies the scale factor value.
- **X Scale Factor**, **Y Scale Factor**, **Z Scale Factor** - For non-uniform scaling, specifies the scale factor value along each axis.
- **Translate** - Moves the selected objects. You define a "from" point and "to" point.
- **From X**, **From Y**, **From Z** - Specifies coordinates of the "from" point.
- **Pick Vertex Position** - Allows you to choose a point in the graphic view to set the "from" coordinate value.
- **To X**, **To Y**, **To Z** - Specifies coordinates of the "to" point.
- **Pick Vertex Position** - Allows you to choose a point in the graphic view to set the "to" coordinate value.
- **Rotate** - Turns the selected objects by a variable amount about a global axis in either a clockwise or counterclockwise direction. The origin of the rotation is always the global origin.
- **Rotation Angle** - Defines the angle of rotation. The rotation angle is limited to a positive real number between 0 and 90 degrees.
- **Axis** - Defines the axis of rotation, which can be the global X, Y, or Z-axis.
- **Rotation Direction** - Defines the rotation direction, which is either **CW** (clockwise) or **CCW** (counterclockwise) about the selected axis using the right-hand rule.
- **Mirror** - Mirrors the selected objects about a plane and optionally, a point.
- **Plane Normal** - Specifies the global axis about which the objects mirror. **Coordinate** - Defines the location along the plane normal axis to serve as the position to mirror about.

Pick Vertex Position - Allows you to choose an object in the graphic view to set the coordinate value on the mirror plane. **Copy and mirror** - Controls whether the selected geometry is mirrored or whether a copy of the geometry is mirrored.

Apply

Applies the operation to the selected objects.

Undo

Reverses the operation by replacing the results with the data before the operation was applied. You can modify a setting, re-apply, undo, and repeat until you are satisfied with the results.

How to Transform Data

1. Click **Transform Data**.
2. Select an object to transform.
3. In the **Operation** box, select an operation to perform.
4. Click **Apply**.

NOTE You can use **Undo** on the ribbon to reverse an operation without exiting the command. For example, you can modify a setting, re-apply, undo, and repeat until you are satisfied with the results.

SECTION 10

Compare Data

Compares a sheet-body in a graphic view against a sheet-body from an existing .sat file.

Because the Geometry Analysis and Repair task modifies geometry data, it is often helpful to compare original and modified geometry to see if the changes are what you intended. You can control the granularity of the comparison, which generates temporary graphics to show the deviation.

Compare Data Ribbon

Sets options for comparing data.

Select the Sheetbody to compare

Select a single sheetbody in the graphic view.

Select .sat file

Select an external .sat file to compare to the selected sheetbody.

Compare

Performs the comparison of the two sheet-bodies. The results are shown in a tabular summary as well as graphically, using colors. See **Deviation Threshold** below for an explanation of the colors.

Points Per Face

Specifies the number of sampling points generated per face of the baseline sheetbody from the external .sat file. This number must be a positive integer between 5 and 25.

Deviation Threshold

Controls the color of the graphic results. The color of the arrows in the graphic view designates the comparison as follows:

- **Green** - Less than the deviation threshold.
- **Red** - Greater than the deviation threshold.

Scale

Controls the scaling of the deviation vectors. The scale must be a positive number.

How to Compare Data

1. Click **Compare Data** on the vertical toolbar.
2. Select a sheet-body in a graphic view.
3. Select a baseline sheet-body from an existing .sat file.
4. In the **Points Per Face** box, enter an integer between 5 and 25. This value is the number of sampling points generated per face of the baseline sheet-body.

5. In the **Deviation Threshold** box, enter a value to control the color of the graphical results. Sample points with a deviation less than this threshold appear in green, while those above the threshold appear in red.
6. In the **Scale** box, enter a value to control the scaling of the graphical results. The scale value can be any positive number.

SECTION 11

Displaying Properties

After you have processed and repaired data in the Geometry Analysis and Repair task, you can view object properties. It is often helpful to view these properties when troubleshooting.

Using the **Select** command on the vertical toolbar, you select the object for which you want to view properties. Then, you click the **Properties** button on the ribbon. The **Properties** dialog box shows general information, attributes, and error descriptions for the selected objects.

An important part of the **Select** command is the **Locate Filter** box that appears on the ribbon. The **Locate Filter** box contains the available, pre-defined filters for the **Select** command. When you choose a filter in the **Locate Filter** box, the software allows you to select only the filtered objects in a graphic view. For example, if you select **Points**, you can select only points in a graphic view.

The Geometry Analysis and Repair task includes these filters:

Construction Graphics

Limits the selection of items to construction graphics.

Points

Limits your selection in a graphic view to points.

Surfaces

Limits your selection in a graphic view to surfaces.

Wires

Limits your selection in a graphic view to wires.

All

Allows you to select any object.

Use the **Inside** fence command to select all objects entirely inside the fence.

Use the **Inside/Overlapping** fence command to select all objects entirely inside the fence and those objects outside but touching the fence at some point.

See Also

Geometry Analysis and Repair Properties Dialog Box (on page 35)

Geometry Analysis and Repair Properties Dialog Box

Displays geometry properties for an object. Click **Print** to print the information on any of the tabs to a file.

General Information Tab (on page 35)
Attributes Tab (on page 35)
Errors Tab (on page 35)

General Information Tab

Displays general information for the selected object.

For a single object (such as a single sheet-body with one face), detailed information appears on this tab. For multiple objects (a sheet-body with more than one face), summary information appears.

Attributes Tab

Displays a list of attributes added to the selected object. The software adds attributes to objects during the import process and during the check process in the **Process Geometry Data**  command.

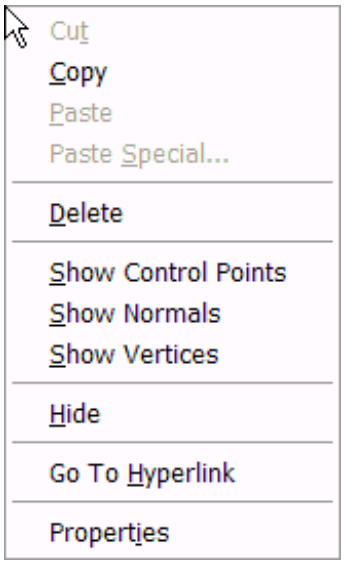
Errors Tab

Displays descriptions of the errors for the selected object. You can check this tab after processing objects in the Geometry Analysis and Repair task.

SECTION 12

Using Shortcut Commands in Geometry Analysis and Repair

The Geometry Analysis and Repair task has a shortcut menu with commands that allow you to see control point meshes, normals, and vertices on the geometry. It can be helpful to see these items as you repair the hull geometry.



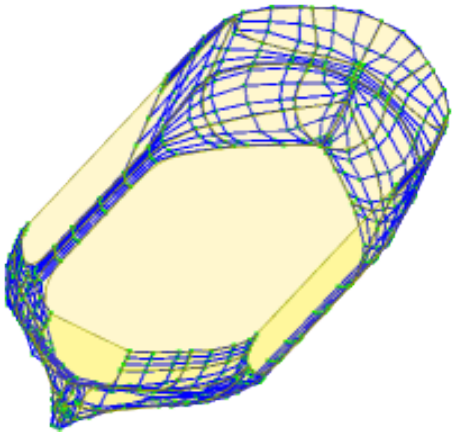
See Also

Show Control Points (on page 36)
Show Normals (on page 37)
Show Vertices (on page 37)
Hide (on page 38)

Show Control Points

Displays the control points and the control point mesh in the graphic view. This command is helpful when you want to see if the control point meshes are aligned. You can access this command by right-clicking an object when no other command is active.

The following picture shows control points and a control point mesh on a hull.

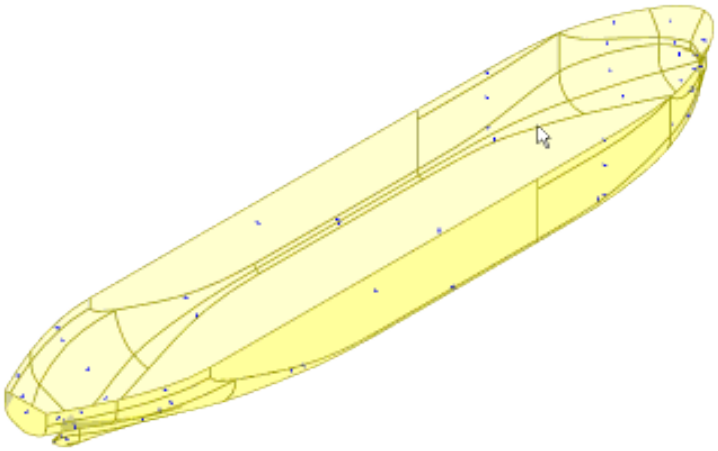


NOTE These control points are not the same as the control points in the Common task.

Show Normals

Displays one surface normal per patch in the graphic view. You can access this command by right-clicking an object when no other command is active.

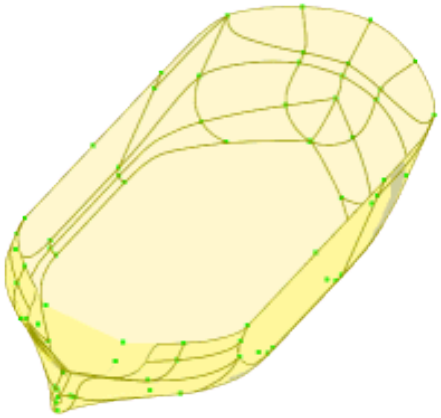
The following picture shows surface normals on a hull.



Show Vertices

Displays vertices in the graphic view. You can select a vertex and view its properties, which include its coordinates. You can access this command by right-clicking an object when no other command is active.

The following picture shows vertices on a hull.



Hide

Hides any control points, normals, and vertices that are displayed in the graphic view. You can access this command by right-clicking an object when no other command is active.

APPENDIX A

General Hullform Requirements

Hullforms must meet several high-level requirements for NPT (non-procedural thickening) and ACIS detailing. The following table summarizes the requirements for hullform models used in Smart 3D.

Source	Description	Checking and Repairing
Smart 3D	Inward- pointing normals	You can visually check the normals in the graphic view. To view the normals, you can use the *Show Normals* (on page 37). In addition, you can use the *Process Geometry Data* (on page 17) to check and repair the normals. To check the normals, use the **Outward Normals** option on the *Check Settings dialog box* (on page 20), and to repair the normals, use the **Orient Normals** option on the *Repair Settings dialog box* (on page 22).
Smart 3D	Single lump sheet-body	You can visually check the sheet-body in the graphic view. In addition, you can use the *Process Geometry Data* (on page 17) to check for multi-lump bodies. On the *Check Settings dialog box* (on page 20), use the **Multi-lump Bodies** option.
Smart 3D	Double-sided sheet-body	You can visually check the sheet-body in the graphic view. If the sheet-body is double-sided, you can see both sides of it when you rotate it. If it is not double-sided, one side disappears when you rotate it. NOTE Convexity errors should be avoided, but are allowable in early design. Use the *Process Geometry Data* (on page 17) to check the continuity. On the *Options dialog box* (on page 19), you can set the Acis check level.
NPT	Non-planar patch edges aligned	You can visually check the patch edges in the graphic view.

Source	Description	Checking and Repairing
		You can also use the *Process Geometry Data* (on page 17). The *Check Settings dialog box* (on page 20) provides the **Non-Aligned Edges** option.
NPT	Uniform control point mesh size for non-planar patches	You can visually check the control point mesh in the graphic view. You can also use the *Process Geometry Data* (on page 17). The *Check Settings dialog box* (on page 20) provides the **Non-Uniform Control Points** option.
NPT	Only planar "trimmed" patches supported	You can visually check the patches in the graphic view. You can also use the *Process Geometry Data* (on page 17). The *Check Settings dialog box* (on page 20) provides the **Trimmed Non- Planar Surfaces** option.

See Also

Geometry Analysis and Repair (on page 7)

Glossary

abaft

Toward the stern of a ship, behind, further aft than.

abstract part

A part that is only defined by a partial specification and that cannot be materially provided by the organization that defines the specification.

Active Template Library (ATL)

Set of class templates and wizards supplied with Microsoft C++ Version 5.0 and later. You can use an ATL when you create ActiveX controls and any other type of object that uses the Component Object Model (COM) model. Using an ATL is generally preferred over Microsoft Foundation Classes (MFC), because the implementations are smaller, easier to use, and more closely tied to the COM model.

aft

Toward, at, or near the stern.

after body

The hull from aft of the midship section.

aftermost

Nearest the stern.

angle

The circular measurement taken from the intersection of two pipes at a turn or branch.

approval state

Recorded state of acceptance of information contained in objects within the database. The approval states indicate a level of confidence in the information stored in the database and govern your ability to alter specific data about a product.

arrangement (accommodation)

Those components of a system arranged in three-dimensional space with accurate dimensional representation for installation. Various types include electrical, HVAC, machinery, outfitting, and piping.

attribute

A single type of non-graphics information that is stored about an object such as diameter or end preparation.

axis

An imaginary line used to define the orientation of a system or object normally defined in terms of an x-, y-, and z-axis. Some 3-D graphic objects have an associated axis used to define the center or axis for rotations.

basic design

Engineering definition of the model and its systems.

bill of material (BOM)

Hierarchical decomposition of a product into constituent assemblies and parts. Specific types of BOMs exist (for example, an EBOM is a bill of material from the point of view of an engineering department; an MBOM is a bill of material from the point of view of manufacturing).

built ships

Complete database of NGC information after completion of the ship contract.

bulkload

The process by which reference data in Microsoft Excel workbooks is loaded into the Catalog database.

catalog

Repository of information about components and materials used in construction. When you use catalog parts in the model, the software places an occurrence of the catalog part in the project. This occurrence is a copy of the actual catalog part.

Catalog database

The database that contains the reference data. Each model database can reference a different Catalog database.

ceiling

Overhead design of the cabin area, including distribution systems for power, water, and ventilation.

chain

A set of continuous and tangent segments.

change history

Process of recording information such as who, when, and why for any given modification.

change management

Software features or manual procedures for managing the consequence of change. For example, software can support a change management feature to report drawings that need updating as a result of a change in a 3-D model.

change propagation

Ability of the software to intelligently modify dependent design information to reflect change in a higher order object.

class

Grouping of individual objects that share some very significant, common characteristics.

class rule check

Verification that the developing design meets the rules of a particular classification society, such as ABS, Lloyd's, or DNV.

Class Rules

Classification Society Design Rules.

classification folder

A folder in the Catalog hierarchy that contains part classes. Classification folders are one level above part classes. The ClassNodeType and R-ClassNodeDescribes sheets in the Microsoft Excel workbooks define the classification folders.

codelist

A set of acceptable values for a particular property that can be referred to by an index number or selected in a combo box. For example, the codelist for the material specification allows you to select from a set of standard entries, such as ASTM A183-F316 Stainless Steel.

commodity code

A user-defined code that provides an index to parts in a catalog.

commodity item

A standard component found in a manufacturer catalog (an off-the-shelf component).

component

Physical part that a feature generates.

concurrent access

Ability of the software to allow multiple users to simultaneously access and modify the design of a model.

consolidated tasks

A collection of tasks run in batch. For example, the software allows you to extract a set of drawings immediately or to schedule the batch extraction for a future time.

constraints

A logical restriction that controls how part symbols ports relate to each other and to reference ports. There are four constraints: parallel, perpendicular, coincident, and distance.

contract

A Work Breakdown Structure object representing a scope of work, usually performed by an external supplier. The contract is related to a project and appears in the Work Breakdown Structure hierarchy.

control point

A point that is used to control the shape of a NURBS curve or surface. Curves have a one-dimensional array of control points, while surfaces have a two-dimensional array.

coordinate

The location of a point along the X-, Y-, or Z-axis.

coordinate system

A geometric relation used to denote the location of points in the model. The most common coordinate system is the rectangular coordinate system, whereby points are located by traversing the X-, Y-, and Z-axes of the model. Normally, coordinate systems have their origin defined as 0,0,0.

cutting plane

A plane that cuts through an object.

damage records

Data relating to the damage and repair of structure or components that occurred during or after construction of a plant.

data interchange

Capability to output the design, or portions of the design, in a standard format for use or movement to another computer software system.

database

Repository for the product model data. The database contains information to describe individual objects in the data model and the relationships between objects as appropriate.

database backup

Process of recording a backup copy of the complete database or the incremental changes after the date that the last complete copy was created.

database break and recovery

Utilities used to restore a database after files are corrupted.

database copy

Functionality to copy large collections of model objects from one design project to another design project.

database management

Functionality related to managing a product model database.

database monitor record

Transactions that occur in order to provide database (DB) recovery after a stop in response with a minimum of lost data.

degree

The highest polynomial factor in the curve or surface mathematical definition. A line is a degree 1 curve, while a cubic B-spline is a degree 3 curve.

design alternative

Difference in a design represented by a separate version. A design alternative can be a new design prepared as a proposed change, or one of several elective options that the builder or customer selects. Each design alternative has an identification assigned so you can uniquely refer to the design alternatives.

design approval log

Record of review and approval of parts of the design.

design data auto input

Automation in loading existing design data into a new design database.

design documents

Drawings, sketches, material lists, procedures, and so forth that are generated during the design phase.

design object

Any object with properties that you can select. A design object can be related to one or more contracts of different types, but related only to one contract of a given type.

design progress check

Analysis of the content of the design to some metric unit that gives an idea of the degree of completion.

design review

Functionality to support rapid viewing of the design and markup of features with comments.

design service

Any general system services related to the design function.

design standard

Feature or object used in plant design that has been determined to the normal or approved way of accomplishing a design requirement. In the context of computer software, the term refers to computer functionality to support standards, not the standard itself.

detail schedule

Lowest level of schedule used to manage and track work progress.

distributed systems

Systems consisting of sequential parts with a distributive characteristic (for example, pipes distribute fluids, HVAC distributes air, cabling distributes power, and structure distributes loads).

distribution systems

Term synonymous and used interchangeably with the term distributed systems.

documentation

Drawings and other records that you must produce to document, obtain approval, or build the design.

drawing tool

Tool that helps in the process of creating, modifying, or manipulating objects. Examples are PinPoint and SmartSketch.

easting

A term that describes an east coordinate location in a coordinate system.

edge

A topological object that represents a trimmed curve bounded by a start and end vertex.

edge distance

The distance from the center of a bolt or rivet to the edge of a plate or flange.

equipment catalog

Catalog of equipment geometry and limited properties that the software uses to identify and visualize equipment and its placement in the model. The catalog is not the source for the total specification and ordering data for the object.

external appendages

External structure attached to the hull, such as the propeller nozzle, shaft struts, bilge keel, and so forth.

fabricate

To cut, punch, and sub-assemble members in the shop.

face

A topological object that represents a trimmed surface bounded by a loop of edges.

face plate

An edge reinforcement type that places a plate or profile at the selected plate edge.

face-to-face

The overall length of a component from the inlet face to the outlet face.

fasteners

Bolts and rivets used to connect structural members.

element

Primitive geometric shape such as a line, circle, or arc.

fence

Boundary or barrier that separates or closes off an area. To surround or close like a fence.

field adjustment

Material added to the neat design geometry of piping or structural parts to allow for fit up in the case that extra material is required due to uncontrolled variance in the manufacturing and construction process.

fire integrity

Deck and bulkhead treatments and fire and smoke blocks for fire control and retardation.

flavor

A different variation of a symbol. Each variation has different occurrence property values.

focus of rotation

A point or line about which an object or view turns.

full penetration weld

A type of weld in which the weld material extends through the complete thickness of the components being joined.

function points

Part of the requirements documentation, function points are the smallest granularity of a requirement statement that describe specific detailed actions that the software performs.

functional block diagram

Schematic representation of a system (piping, electrical, ventilation) showing system parts and their relationship. You use symbols to represent equipment and components. A connecting network of lines illustrates their relationship. Taken together, the symbols and the network illustrate the function of the system.

furnishings

Parts such as movable articles and fittings that normally are not associated with a system (for example, a chair).

generic specific

Object that is parametrically defined or defined to suit a family of specific parts (for example, International Standards parametrics). For example, a 100 - 200 gpm pump in the catalog can provide a general shape to appear in the model until a specific object has been identified. See also specific and specific object.

GUIDs

Acronym that stands for Globally Unique Identifiers. The software automatically creates the GUIDs sheet in the Excel workbooks when you create the Catalog database and schema. The purpose of storing GUIDs within Excel workbooks is to help you keep track of what has been loaded into the database. Storing GUIDs also helps to avoid the situation in which a replacement Catalog database causes existing models to become invalid.

host location

The first location created for a Site. This host location is defined when the Database Wizard creates the Site database.

host server

The database server on which the Site database was created using the Database Wizard. Alternatively, if it is a restored database set, the Host Server is the database server where the Site database is restored. The Host Server in a Workshare environment contains the origin for the Site, Site Schema, Catalog, and Catalog Schema databases. Consequently, most Project Management and reference data work must take place at the Host.

hull applicability versioning

Capability within the database that allows the database to have a single model for a class of ships that are nearly identical (sister ships) while recognizing and managing the different parts in each ship of the class. The concept considers that alternate configurations are created for a state purpose (owner change order, new vendor, design improvement, and so forth). In addition, this concept relates the geometry and associated geometry to that purpose, and provides an extraction method from the database for viewing or manufacturing the base ship with the appropriate variant of the design authorized for that particular ship hull.

hull form generation development

Set of ship lines or surfaces that meet ship requirements for speed, powering, fuel rate, and cargo, and any special limitation of draft, beam, or length. At this stage, lines should be adequate for model tests.

hull structure design

Functions related to creating the structural design and manufacturing attributes of the hull and longitudinal and transverse structure of a ship.

initial design

Early stage of design work, generally before contract, used to estimate construction costs and provide a rough concept of the intended plant. Contains information relating to a plant created during its initial (concept) design period.

initial structural plan

Principal structural plan for the plant; also called a construction profile.

instantiation

Occurrence of a catalog object at a specific geometric location in the model.

interference checking

A process that identifies possible collisions or insufficient clearance between objects in the model.

job order

Industrial authorization for accomplishing work; synonymous with a work order.

joiner

Non-structural bulkheads, and trim and built-in furnishings.

kinematics analysis

Analysis of mechanical motion.

ksi

Kips per square inch.

leg length analysis

Preferred term is welding length analysis.

library

Resource of reference information that you can access in developing a plant design.

life cycle database

Information developed to assist in the maintenance and modernization of delivered plants.

link

Way to store information about another file in your document. You can update a link so that changes in the file appear in your document.

lintel

A horizontal member used to carry a wall over an opening.

load group

A grouping in which all components feature uniform load limits and stress safety characteristics. For example, if a pipe clamp from load group 5 has a maximum nominal load of 20kN, then so does a threaded rod from load group 5.

location

A Location is defined by three user-defined inputs: 1) a unique name, 2) a unique name rule ID, and 3) the server where the Site databases reside for that Location. A Location is defined and created when the Site database is created using the Database Wizard. Additional Locations can be created in the Project Management task. Each Location is a Site-level object, thus other Plants within the same Site collection can use the Locations when the Plants are configured for Workshare.

logical member

An object in the model used to represent the design topology.

machinery

Major pieces of equipment installed in a plant.

macro

A sequence of actions or commands that can be named and stored. When you run the macro, the software performs the actions or runs the commands. You can create the macros in Visual

Basic or other OLE-aware programming applications. Some of the other OLE-aware programming applications are Visual Basic for Applications, Visual C++, and so forth.

maintenance envelope

A rectangular box around the part for clearance during maintenance operations.

maintenance parts

Required material for depot or on-board repair or overhaul of equipment, as determined by engineering study. Generally at a level below the purchased construction object of the plant.

maintenance records

Records of breakdown, repair, and overhaul of equipment.

material analysis

Analysis of a completed design work for extracting detailed material requirements; also called material lists.

material list

An option category that controls the format and content of the bill of materials.

methods

Objects in the database that describe the manufacturing methods to the component parts of a plant.

move from point

Starting point for an action. For example, when you move an equipment object, the Move From point determines the point of origin for the move.

move to point

Ending point for an action. For example, when you move an equipment object, the Move To point determines where you want the move to stop.

MTO neutral file

A non-graphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

natural surface

A surface without a boundary curve.

node

- One of the set of discrete points in a flow graph.
- A terminal of any branch of a network or a terminal common to two or more branches of a network.
- An end point of any branch or a network or graph, or a junction common to two or more branches.

northing

A term that describes a north coordinate location in a coordinate system.

nozzle

A piping connection point to a piece of equipment.

nozzle standout

The shortest allowable distance between the connection point of a nozzle and the start point of a turn on the leg connected to the nozzle.

NPD (Nominal Piping Diameter)

The diameter of a pipe.

object

A type of data other than the native graphic format of the application.

occurrence (of part or equipment)

Instantiation of a part of equipment in the model that refers to the part library; an instance of a specific object. The design can be built several times, and therefore the occurrence can apply to more than one hull. Typically, an occurrence points back to a specific object, either for its complete definition, as in the case of a particular valve, or for its made from material, as in the case of a steel plate part cut from sheets. Thus, when a designer selects a component from the catalog and places it at a location in the space of the plant, the software creates an occurrence of that object in the plant design.

occurrence property

A characteristic that applies to an individual object in the model. Occurrence properties are designated with 'oa:' in the reference data workbooks. You can view and modify occurrence properties on the Occurrence tab of the properties dialog boxes in the software. Depending on the object, some occurrence properties are read-only.

origin

In coordinate geometry, the point where the X-, Y-, and Z-axes intersect.

origin point

The point at which the coordinate system is placed, providing a full Cartesian coordinate system with positive and negative quadrants. Points are placed at coordinates relative to the origin point, represented by the X, Y, and Z values.

orthogonal

The characteristic of an element consisting completely of elements positioned at 90-degree angles. A square is an orthogonal element.

orthographic

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

P&ID

Diagram that shows the topology, functional components, and special requirements of a piping system; generally represents the engineering design of the system.

package

Set of closely related classes. (UML)

painting

Computation of paint surface and recording of paint system requirements.

parameter

A property whose value determines the characteristics or behavior of something.

part class

A group of similar objects. You can define part classes in the Excel workbooks. A part class can have multiple parts. For example, a heat exchanger part class can contain heat exchangers with different dimensions.

part number

Unique identifier of a part.

PDS (Plant Design System)

A comprehensive, intelligent, computer-aided design and engineering application for the process, power, and marine industries. PDS consists of integrated 2-D and 3-D modules that correspond to engineering tasks in the design workflow.

physical occurrence

Unique specific object that has traceability and is the physical manifestation of an occurrence object. A physical occurrence applies to one and only one hull. It is a version of its occurrence object with as-built or as-modified differences included and has a serial number or lot number.

PinPoint

Tool that allows you to place, move, and modify elements with precision, relative to a reference point.

principle of superposition

The principle that states that the stresses, strains, and displacements due to different forces can be combined. This principle is only valid for linear analysis.

product

Data objects that describe the components of a ship and any corresponding properties. An individual object or part (or its representation in the product model) that may be installed in the ship. Examples of individual products include objects such as a coffee urn, a light fixture, a piece of pipe, a piece of ventilation duct, a radar display console, a bulkhead plate, and a structural profile stiffening a bulkhead.

Product Data Management (PDM) System

Software intended to manage both product data and documents associated to the product data. Functionality typically includes: object-based data modeling tools, user administration, business rules, and document management. Document management typically includes document editing or reviewing, document mark-up or redline, document storage, and full-text retrieval.

product structure

Hierarchical breakdown or decomposition of a product into constituent parts, volumes, or units. (For example, a bill of material is one possible type of product structure.)

production planning

Functionality associated with the work breakdown and sequence of the construction of a plant.

promotion

Process of associating approval state with a product version. A product version begins its existence at a working approval state. When the version is at some level of maturity, its approval state is elevated to a higher approval state (that is, promoted). Then, further changes must be carefully controlled and generally require the data set demoted to a working state. One or more promotions can occur successively higher approval states (between working and approved) to represent various intermediate levels of review or progressive approval.

query select sets

Set of objects that are selected in a query or queries on the database.

reference data

The data that is necessary to design plants or ships using the software. Reference data includes graphical information, such as symbols. It also contains tabular information, such as physical dimensions and piping specifications.

resource estimation

Rough estimate of material, manpower, and facility utilization for the design and construction of the plant.

route

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

satellite server

The database server where the replicated databases reside for Workshare. The Satellite Server is not used unless Workshare is activated.

schema

A database that creates the structure of another database. For example, a schema specifies the queries, tables, fields, and data types in a database.

schema update utility

Functionality used to assist in processing existing product models to an updated database structure after you modify or add to the database structure.

sheetbody

A topological object that represents a collection of faces joined along their common edges (stitched).

shell structure

External portion of the surface of the plant.

ship

A collection of modeled objects that can be simultaneously displayed and edited in a workspace. A Ship points to a Catalog (optionally shared with other Ships). Access control is managed at the Ship level.

site

The top level in the Project Management hierarchy. A Site configuration may contain several Catalogs, each shared by multiple Plants.

site administrator

Person responsible for managing the standards and general parameters for a given plant site within a Site database.

site setup

Functionality associated with establishing a new plant site or hull for design development.

sketch and trace

User interface for rough definition of a required design feature that typically works in a 2-D mode.

specials

An option category that allows you to control specialized calculations for equipment trim, repeatability, and center-of-gravity.

specifications

Contracted requirements for the plant.

steel outfitting

Internal structural elements of a ship that are required to meet a local requirement such as foundations, non-structural bulkheads, walkways, and so forth.

stern frame

Casting and structure that support the rudder and shaft opening.

stud

A bolt, threaded on both ends, used to connect components.

suspended floor

A concrete floor system built above and off the ground.

swash bulkhead

A longitudinal or transverse nontight bulkhead in a tank that decreases the swashing motion of the liquid contents. A plate in a tank that has this same effect but that does not extend to the bottom of the tank is called a swash plate.

symmetric node

Type of vertex on a curve. A curve with a symmetric node has the same curvature on each side of the node. A handle can be attached to a symmetric node for editing.

system

A conceptual design grouping that organizes parts in hierarchical relationships. A system represents a functional view of the model and includes information such as system name, type, properties, and design specifications for the objects assigned to the system.

tag number

User-specific, unique number assigned to an object (for example, CV-101 for a control valve, HE-2002 for a heat exchanger).

target point

The origin for coordinate measurements displayed by PinPoint. You can position the target point anywhere on the drawing sheet or view.

tolerant geometry

A type of ACIS geometry - either an edge or a vertex - that is outside the tolerance for ACIS and requires special handling.

transverse

At right angles to the fore-and-aft center line.

transverse frames

The athwartship members that form the ribs of the ship.

trim

The difference between the forward draft and the aft draft.

trimmed surface

A surface whose boundary is fully or partially inside the "natural" geometric definition of the surface. Some or the entire control polygon extends outside the face boundary.

trunk

Feature that quickly reserves space for the distributive systems and other systems that have a path. Along the trunk are stations that define the cross section and identify part or system membership.

tumble home

The inboard slope of the side of a ship, usually above the designed waterline.

unit/module modeler

Facility of the system to structure collections of equipment and components into a single identifiable object.

user attributes

A customized property in the reference data. The Custom Interfaces sheets in the Excel workbooks define these properties. You can list the customized properties on the individual part class sheets.

version control

Ability of the system to manage multiple versions of a single part of the design. Version control should support conditional analysis and promotion status, as well as alternate design features among hulls within a plant site.

vertex

A topological object that represents a point in the three-dimensional model.

vertical keel

A row of vertical plates extending along the center of the flat plate keel.

viewset

Set of objects (usually a subset of the entire database) that a view operation uses. Membership or lack of membership for any object in a viewset does not affect the actual stored representation of the object, but only its availability or desirability for viewing in the current scenario.

water line

A line parallel with the base line that depicts the water.

watertight door

A door that when closed prevents the passage of water.

weather deck

A deck exposed to the weather.

weathertight door

A door that when closed prevents the passage of rain and spray.

weight and CG analysis

Routines that compute the weight of commodity materials as configured in a given design (for example, plate and pipe) and determine total weight and center of gravity (CG) for a collection of material and equipment, as well as the complete plant.

welding

Weld requirements for joining materials. Welding length analysis is the calculation of required weld dimensions; also called leg length analysis.

windlass

The machine used to hoist and lower anchors.

wirebody

A topological object that represents a collection of edges jointed at their common endpoints.

wizard

Software routine attached to an application that provides guidance and expert help to you to complete one of the functionalities of the application.

work content

Estimation development of metrics from the database that relates to the work hour content of the various construction units.

work order

Plant authorization for completing work; synonymous with a job order.

working plane

The available 2-D plane of movement for endpoint selection.

workset

Set of objects (usually a subset of the entire database) used in an interactive change, add, or delete operation. Membership or lack of membership for any object in a workset does not necessarily affect the actual stored representation of an object. However, you can change or delete an object in a workset that also results in a change or deletion of the stored object. Similarly, when you add a new object (not currently stored) to a workset, the software also adds the object container.

workspace

Area that represents the portion of the model data needed to perform the intended task and includes the user modeling settings.

workspace document

Document into which you can extract a portion of the model data for a user task.

Workspace Explorer

Tree or list representation of objects in your workspace.

Index

G

H

I

J

K

L

M

N

O

P

Q

R

S

T

U

V

W